EECS 373
Design of Microprocessor-Based Systems

Robert Dick
University of Michigan

Lecture 12: Memory and PCBs

16 February 2017

Slides inherited from Mark Brehob.

1

Outline

- Context and review
- Memory
- PCB design

2

Context and review

- ADC and DAC operation and error
  - Speed / accuracy / monotonicity trade-offs.
- Prototyping
  - Breadboards.
  - Soldering.
  - Wire wrap.
- Memory
  - Flash
  - Speed, functionality, reliability trade-offs.
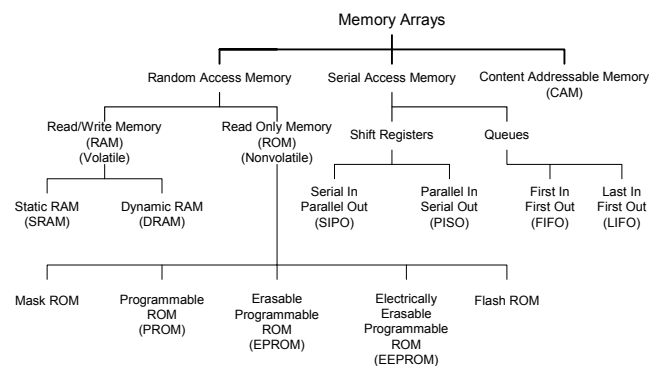
3

Outline

- Context and review
- Memory
- PCB design

4

Memory: why?

- You'll be dealing with this a lot in your projects.
- A little review now can save you trouble later.

5


Memory array types

6

Nonvolatile memory types

- Ferroelectric RAM: polarization changed by electric field. DRAM/flash-like. Electical polarization instead of floating gate. Reading → current pulse on write? 150 ns. 10^{12} cycles.
- Phase change memory: amorphous (off) vs. crystalline (on). 5ns to 100 ns write. ns read. 10^8 cycles.
- Magnetoresistive RAM. ns read. Slightly more for write. Many cycles.
- Programmable metallization cell: redox filament. 10s of ns read, write. 10^{10} cycles.
- Flash: ns read, ms write.
- EEPROM: ns read, ms write.
- SRAM/DRAM: ns read and write.
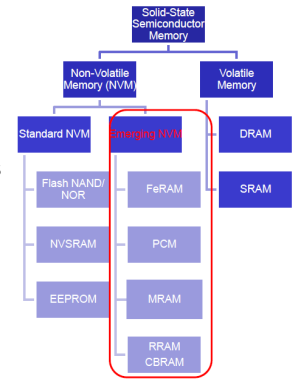
Floating gates

- Write: hot-electron injection or Fowler-Nordheim tunneling.
  - High voltage on control gate >> operating voltage
  - Electrons are trapped in the floating gate.
  - Will not discharge for many years.
- Erase? Fowler-Nordheim tunneling.

control gate
floating gate
source   drain
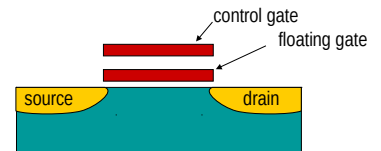
- Read by seeing whether it acts like a transistor or a wall.
- Tend to self-destruct after 10^5 write/erase cycles.

Outline

- ~~Context and review~~
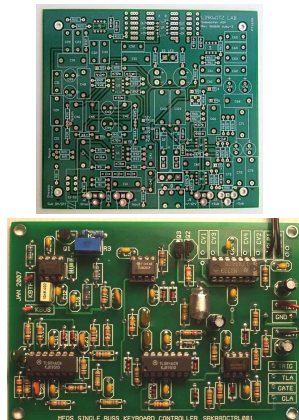- ~~Memory~~
- PCB design

PCBs: why?

- Even if you aren't making one for your project, need to understand how they work for debugging / reverse engineering.
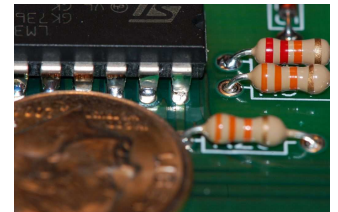
Printed circuit board design

- Physical support.
- Electrical connections.
  - Traces have restricted dimensionality.
  - Very thin, high resistance.
  - Holes/vias and pads.
  - Rework is hard.

http://www.linkwitzlab.com/Pluto/supplies-subw.htm,
http://www.musicfrommouterspace.com/analogsynth/SINGLEBUSSKEYBOARD2007/SINGLEBUSSKEYBOARD2007.php
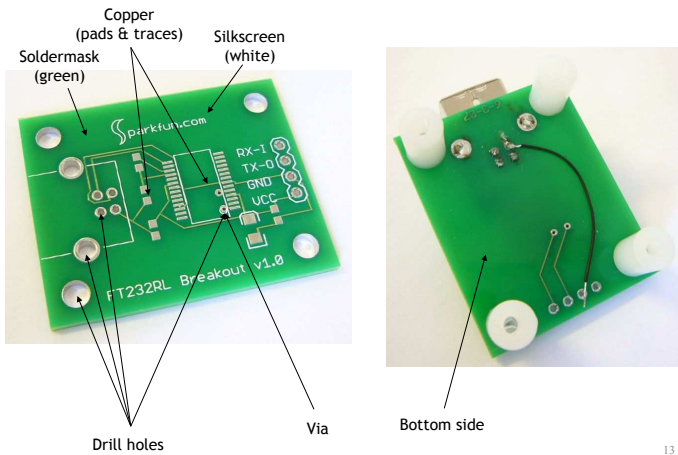
Basic terminology

- Interconnects: traces.
- Inside of a given "layer" traces which cross are electrically connected.
- Can have multiple conductive layers by stacking/bonding boards.
- Through-hole: Having holes in the PCB designed to have pins put through the holes.
  - Contrast with surface mount where device goes on top.
- Surface mount.

Parts of a PCB

Copper
(pads & traces)

Soldermask
(green)

Silkscreen
(white)

RX-I
TX-0
GND
VCC

FT232RL Breakout v1.0

Drill holes

Via

Bottom side

13

Vias

- Sometimes you need to connect traces on layers.
  - Use a via: a plated through hole
    - Generally smaller than a through hole for a pin.

Clearances

- There will be space between the traces, other traces, and plated holes.
  - You need to meet the requirement of the manufacturer.

The layered construction of a PCB: a six layer board

Total Height (62.853mil)

Top Layer
MidLayer1

MidLayer2
GND (GND)

MidLayer3
VCC (Multiple Nets)

MidLayer4

Bottom Layer

Prepreg (5mil)
Core (10mil)
Prepreg (5.236mil)
Core (10mil)
Prepreg (5mil)
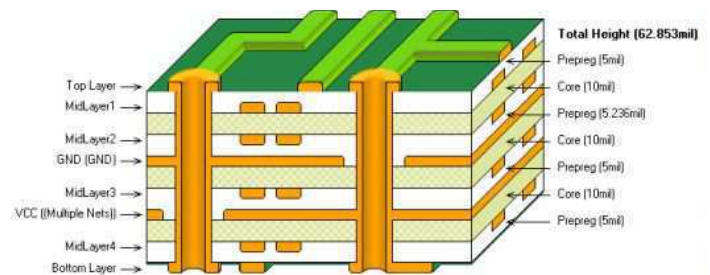Core (10mil)
Prepreg (5mil)

Figure from altium.com

16

What do do with layers?

- Mostly orthogonal routing layers.
- Ground planes to increase local power supply capacitance and minimize resistance.
- Power plane for similar reason.
- More layers → higher cost.

How to design PCB

1. Create schematic

2. Place parts

3. Route interconnect

4. Generate files

Step 1: Create schematic

- The first thing you want is something that looks like a textbook circuit diagram. It just shows the devices and how they are connected.
  - Sometimes you will worry about pinouts here (say when working with a microprocessor maybe).
  - But usually you don't.
- No notion of layout belongs here, although the schematic connectivity will influence layout.

Example schematic
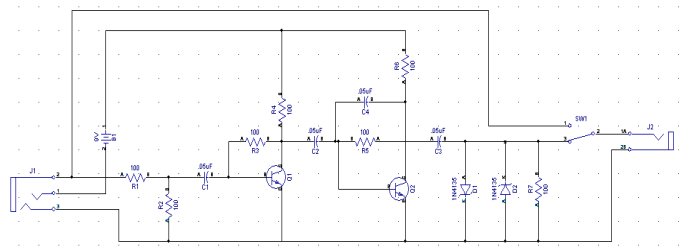
Why a schematic?

- Communication and formalization are main goals.
  - This is probably what your sketch on paper would look like.
  - You can find and fix bugs more easily here than the PCB layout.

Step 2: Place parts

- You need to place the patterns on the board.
  - You need to not overlap them to that the components can actually fit on the board.
  - You want to leave room for the traces to connect everything.
- This is very much an art form.
  - In fact you will find people who rant about "sloppy" or "unprofessional" placements.
- Some tools will do this for you.
- Sometimes they screw up.

Patterns

- Once you know what it is you want to build, you need to figure out how to lay it out on the board.
  - You need to know how big each piece is, and where the holes need to be placed.
- Each device has a pattern.
  - You will occasionally need to create a pattern.
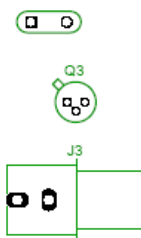
Step 3: Route interconnect

- A route is a connection between devices.
  - It may consist of multiple traces
- There are design rules which include:
  - Minimum trace width
  - Minimum spacing between traces and holes
  - Minimum spacing between holes and holes.
- Rules vary by manufacturer.
- Units vary my manufacturer.

Issues of measure

- PCB designers use odd terminology.
- A "thou" is a thousandth of an inch.
- A "mm" is a millimeter
- A "mil" is a thousandth of an inch.
  - Thou is generally preferred over mil to avoid confusion, but most tools/vendors use mil.

Trace width

- In general most PCB manufactures seem to have trace-width minimums of 6-10 thous.
  - Most are willing to go smaller for a price.
- A rule of thumb is to use a 50 thou minimum for power/ground and 25 for everything else.
  - This is to drop the resistance of the traces.
  - In general you are worried about heat dissipation
- There are lots of guidelines for width/power but in general you are looking at:
  - A 10cm trace needs to be 10 thou wide if it will carry 1 amp.
 - 5 amps at 10cm would require 110 thou.

Trace width continued

- The problem with wide traces is that they are hard to route.
  - In particular you might wish to go between pins of a device.
- One solution is to be wide normally and "neck down" when you have to.
  - This is more reasonable than you think.
   - Think resistors in series.

Rat's nest.

- Rat's nest shows device placements and connections but not routing.
  - Automatically generated for you.
  - Sometimes before placement, sometimes after.
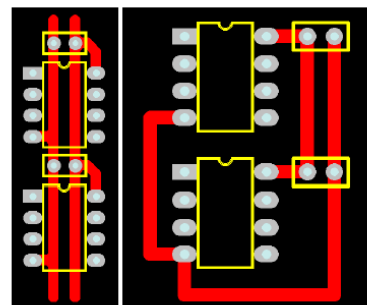 - Varies by tool.
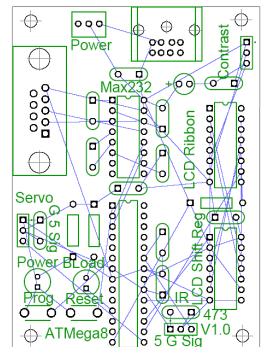
Routing for real

- You can use an autorouter to route your traces
  - Some people hate these as the design will be "ugly".
  - Saves a lot of time.
  - Oddly, not always as good as a person can do.
    - But much faster.
- Still generally need to do some (or all) of the routing by hand
 - Very, very tedious.
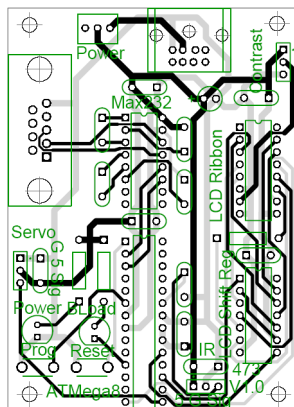
Routing quality

An example of GOOD power routing (Left) and BAD power routing (Right)

Step 4: Generate files

- Once the design is done, a set of files are generated.
  - Each file describes something different.
    - Copper on a given layer.
    - Silkscreen.
    - Solder mask.
  - Most files are in "Gerber" format.
    - Human-readable (barely) ASCII format.
    - Has commands like *draw* and *fill*.
  - Drill files are a different format called Excellon.
    - Human-readable (barely) ASCII with locations and diameters for the holes.
- Generally you zip all these files up and ship them as a single file to the PCB manufacturer.
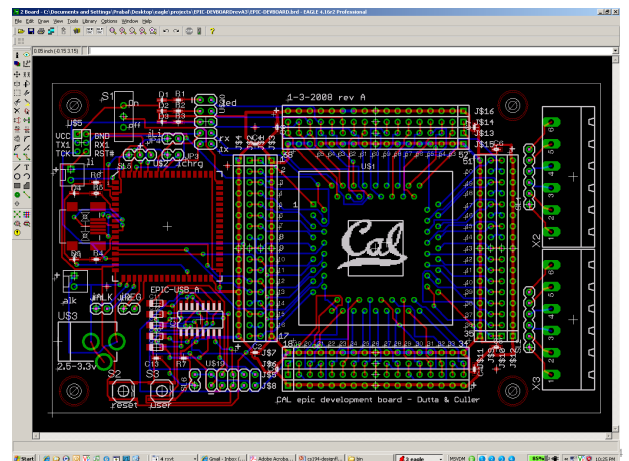 - Often a good idea to include the design file(s) too.

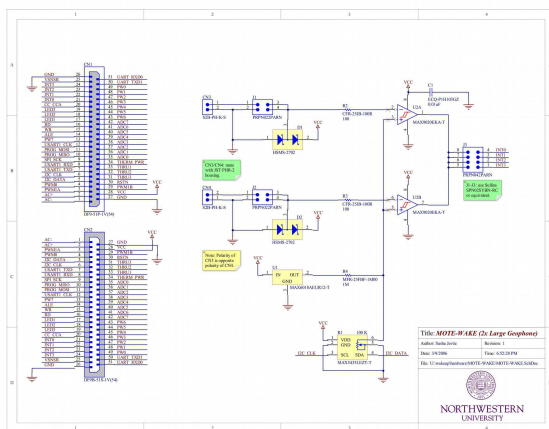## The schematic captures the logical circuit design
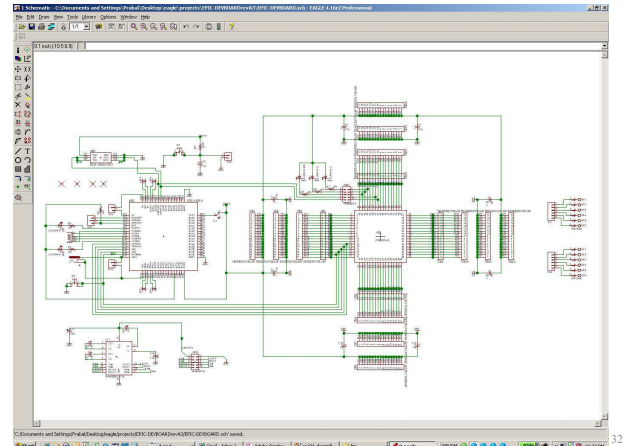
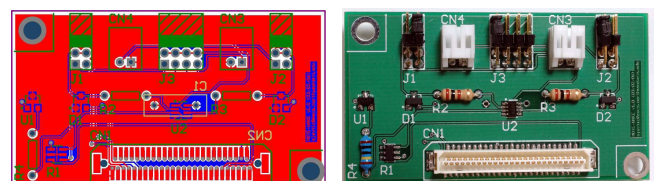## Floorplanning captures part locations
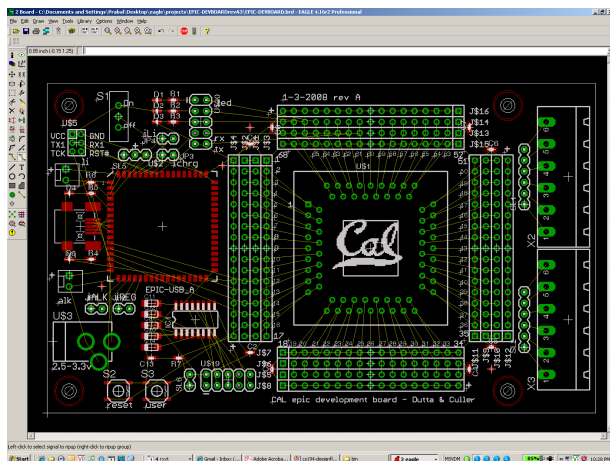
The auto-router places tracks on the board, saving time

Another design, all the way to production

Another simple design, all the way to production

- Simple design that solved a hard problem.
- Deployed at many sites around U.S.

Not a simple design

- Note component density
- Can mount components on each side.
- Relationship between PCB layout, pinouts, and external components important.
  - LED.
  - Battery.
  - Others, e.g., big inductors.
- Form (and board shape) follows function.
- RF subsystem physical design tricky.

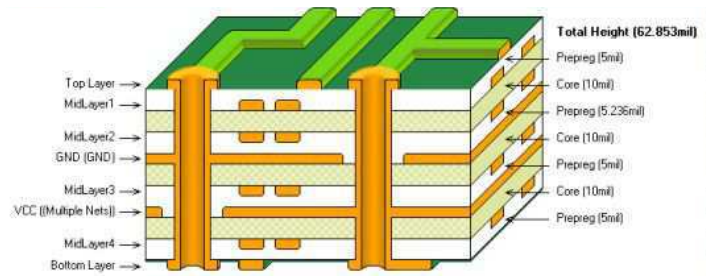## The layered construction of a PCB:
A six layer board

Figure from altium.com

Doesn't need to be expensive / complex

- Can CAD/CAM mill away solid Cu layer.
- Can use lithography.
  - Photoresist.
  - Mask (can print with laser printer).
  - Projector.
  - Etchant (many are dangerous to breathe and touch).
  - Safe way to dispose of Cu-containing solution.